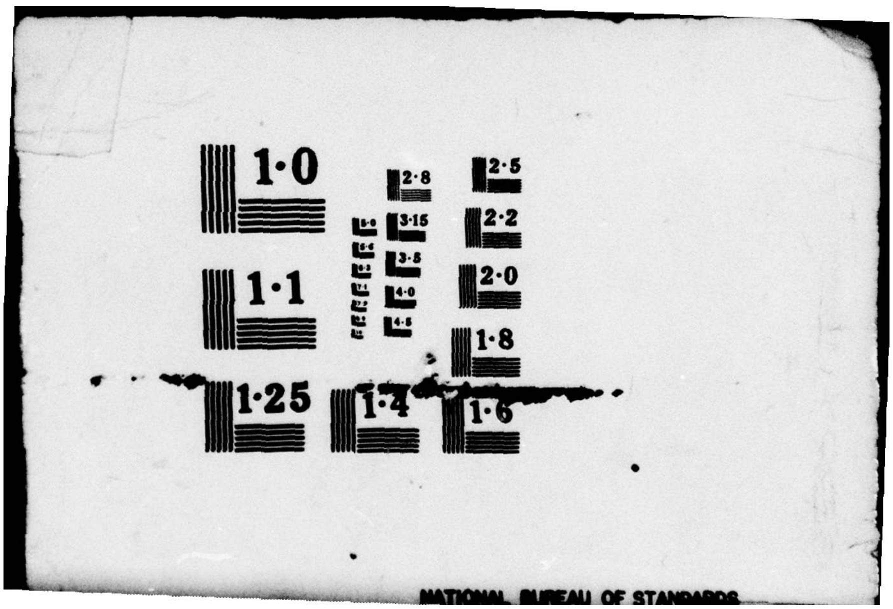
AD-A073 057 SCIENTIFIC RESEARCH ASSOCIATES INC GLASTONBURY CONN* F/G 21/5
COMPUTATION OF DISCRETE HOLE FILM COOLING FLOW USING THE NAVIER--ETC(U)
JUL 79 J P KRESKOVSKY, W R BRILEY, H MCDONALD F49620-78-C-0038
UNCLASSIFIED R79-910002-1 AFOSR-TR-79-0935 NL

| OF |
AD
A073057

END
DATE
FILMED
9 79
DDC


1·0
2·8
2·5
3·15
2·2
3·5
1·1
4·0
2·0
4·5
1·8
1·25
1·4
1·6
NATIONAL BUREAU OF STANDARDS

AFOSR-TR- 79-0935

Report R79-910002-1

LEVEL III

COMPUTATION OF DISCRETE HOLE FILM COOLING FLOW
USING THE NAVIER-STOKES EQUATIONS

F49620-78-C-0038

J. P. Kreskovsky, W. R. Briley [redacted] H. McDonald
Scientific Research Associates, Inc.
P.O. Box 498
Glastonbury, Connecticut 06033

A073057

July 1979

17 p.

Prepared for

Air Force Office of Scientific Research

392 929

79 08 22 002

TABLE OF CONTENTS

ABSTRACT

An investigation is being performed to develop an analysis and computational procedure for predicting the flow field which results from discrete hole film cooling. This study represents the first step in the development of a computer code capable of predicting aerodynamic loss levels and heat transfer levels associated with discrete hole film cooling on gas turbine blades. The specific problem being considered here is that of predicting the flow and temperature fields resulting from injecting a row of hot or cold jets into a cross flow through a flat plate. The approach is based on numerical solution of the three-dimensional compressible Navier-Stokes equations, and utilizes a zone embedding procedure to limit the computational domain to the immediate vicinity of the coolant injection. The zone embedding is accomplished through the use of interactive boundary conditions at inflow and outflow boundaries, which are compatible with flow behavior outside the computational region. In this initial phase of the investigation, techniques for applying interactive boundary conditions were examined for a simplified model problem consisting of normal injection through a two-dimensional slot. The zone embedding approach was found to be successful in limiting the computational region, and the use of specified total pressure as an injection boundary condition was found to produce a nonuniform inlet velocity distribution consistent with interaction between the inlet jet and the free stream. During the next phase of the investigation, these zone embedding techniques will be refined and applied to the three-dimensional discrete hole film cooling configuration.

INTRODUCTION

To achieve higher turbine efficiencies, designers have been forced to deal with high turbine inlet temperatures. As a consequence, structural limitations of the materials used in the production of turbine blades require the introduction of cooling to ensure that the blades remain intact when exposed to this high temperature environment for prolonged periods of time. To this end, various cooling schemes have been devised. Perhaps one of the more idealistic methods of cooling turbine blades is through transpiration. In transpiration cooling a portion of the turbine blade is constructed with a porous medium through which cooling air is forced. This process results in a more or less continuous film of cool protective fluid being injected between the hot gas main stream and the blade surface. Unfortunately, transpiration cooling suffers both from structural and hole clogging problems when actually used and thus it does not represent a realistic cooling configuration.

An approach which approximates this idealized transpiration cooling scheme, but which does not suffer from hole clogging or structural problems, and which in addition is more practical from a manufacturing viewpoint, is discrete hole film cooling. In discrete hole film cooling, a single row, or multiple rows of staggered finite diameter holes are used to approximate a porous surface. The cooling fluid is then injected through these holes to form the protective cool fluid layer. Obviously, the number of parameters which could be varied to obtain a configuration which optimizes heat transfer and aerodynamic loss characteristics is not trivial. Hole shapes and patterns, injection angles and rates, temperature ratios between the hot main stream and the injected fluid, as well as the cooling fluid used, are only some of the possible parameters which can be varied. Thus, rather extensive and costly experimental programs are currently required to determine an optimum configuration. Therefore, it is apparent that a computational procedure which would allow a screening of many proposed configurations without experimental testing would be of great value to the turbine blade designer. In considering the development of such a computational procedure, it should be recognized that even in the seemingly simple case of a single jet injected

normal to the main stream through a flat plate the resulting flow field is highly three-dimensional and viscous effects play an important role. Additionally, under certain conditions the injected coolant jet may cause sufficient blockage effects to separate the flow on the coolant jet lee side. Thus, if such a flow is to be computed in a manner consistent with the anticipated separation region, it is necessary to solve the full three-dimensional Navier-Stokes equations. Use of the full Navier-Stokes equations also permits treatment of interaction between the injected flow and the freestream.

METHOD OF SOLUTION

Background

The complex three-dimensional nature of the flow field resulting from injection of a coolant into a cross stream has prevented adequate predictions by analytical methods in most cases. The flow field surrounding injection is such that injecting more coolant does not necessarily provide more cooling to the surrounding flow surfaces. When the blowing rate is low enough and/or the injection angle is shallow enough, the jet attaches to the wall either immediately or within a short distance downstream of the hole. This case represents a good film cooling configuration. At higher blowing rates the jet may penetrate into the main stream and diffuse before attaching to the wall as a cool layer; this obviously is an inefficient configuration.

Colladay and Russell (Ref. 1) performed a flow visualization study of the discrete hole film cooling problem in several configurations, which serves to demonstrate the three-dimensional nature of the flow and the importance of the injectant-freestream interaction. In this study, they found that normal injection provides the least effective film cooling configuration. For normal convection, the injected coolant detaches from the surface and a pair of counter rotating vortices are formed, even at low injection rates. These vortices promote mixing and entrainment of hot freestream fluid, thus reducing the film cooling effectiveness and also increasing aerodynamic losses. Bergeles, Gosman and Launder (Ref. 2) also noted a small region of reversed flow behind a jet injected normal to the freestream which indicates flow detachment. Although this region extended only one or two diameters behind the jet, it is expected to have a significant influence on heat transfer. Colladay and Russel also investigated injection inclined with the wall and, as might be expected, found that the coolant remained attached

---

[1] Colladay, R. S. and Russell, L. M.: "Streakline Flow Visualization of Discrete-Hole Film Cooling With Normal, Slant, and Compound Angle Injection", NASA TN D-8248, September, 1976.

[2] Bergeles, G., Gosman, A. D. and Launder, B. E.: "The Prediction of Three-Dimensional Discrete-Hole Cooling Processes: I-Laminar Flow", ASME paper 75-WA/HT-109, 1975.

to the surface at much higher blowing rates. This provides a significant increase in the film cooling effectiveness while maintaining low aerodynamic drag. Finally, Colladay and Russell made observations of lateral or skewed injection not aligned with the freestream, and found that separation was delayed until much higher coolant flow rates, although entrainment of freestream fluid was increased. These experimental observations emphasize the importance of interaction between the injected fluid and the freestream.

Previous Analytical Work

Analytical methods directed specifically at film cooling have been developed by a number of investigators and are reviewed briefly in Ref. (3). For example, Ericksen, Eckert and Goldstein (Ref. 4) devised a method for normal injection based upon point and line energy source representations of the jet. Although the method yielded qualitatively reasonable results for film cooling effectiveness, it requires a high degree of empirical information and the authors recommend that the method be used mainly as an interpolation formula rather than a prediction method. Other investigators have used two-dimensional finite-difference boundary layer procedures to predict film cooling effectiveness (see, for example, Refs. 3 and 5). Although these methods are capable of predicting the film cooling effectiveness with a fair degree of accuracy, they are either limited to two-dimensional flows without separation which result from slot injection, or they assume no spanwise variation in the flow. In addition, these methods generally must start downstream of the injection point and cannot supply detailed information

---

[3] Crawford, M. E., Kays, W. M., and Moffat, R. J.: "Heat Transfer to a Full-Coverage Film-Cooled Surface with 30 Degree Slant-Hole Injection", Thermoscience Division, Dept. Mech. Engr. Stamford University, Report HMT-25, May, 1976.

[4] Ericksen, V. L., Eckert, E. R. G., and Goldstein, R. J.: "A Model for Analysis of the Temperature Field Downstream of a Heated Jet Injected into an Isothermal Crossflow at an Angle of 90^{o} ", NASA CR-72990, 1971.

[5] Shamroth, S. J. and McDonald, H.: "Calculations of Film Cooling, Wall Transpiration, and Heat Transfer with a Finite-Difference Boundary Layer Procedure", United Aircraft Research Laboratories Report N212634-1, January, 1973.

regarding the injection-freestream interaction. The only previous study which treated the discrete hole film cooling problem as a three-dimensional flow is that of Bergeles, Gosman, and Launder (Ref. 6), who applied the approximate calculation procedure of Pratap & Spalding (Ref. 7) for the case of laminar flow. This calculation procedure neglects streamwise diffusion and employs iterated forward marching solution of three approximate momentum equations. The procedure begins with a "guessed" pressure field and performs iterated forward marching sweeps of the three-dimensional flow field, solving the approximate momentum equations and utilizing various strategies to modify or correct the pressure field, so as to improve the continuity balance. Since this method is a forward marching procedure which requires the identification of a primary flow direction, it does not appear to be well-suited for high coolant injection rates where the primary flow direction varies considerably near the jet exit plane, or when there is separation and reversed flow downstream of the injection hole. In addition, whatever the merits of the various approximations which are introduced, the computational effort required for iterated forward marching methods tends to be comparable with that required for solution of the Navier-Stokes equations without approximation, using recently developed efficient split LBI schemes (e.g., Ref. 8). Since solution of the Navier-Stokes equations relieves restrictions regarding primary flow direction and reversed flow regions, as well as providing sufficient generality to deal with variable injection rates, angles, and other flow parameters, this is the approach being taken in the present investigation.

---

[6] Bergeles, G., Gosman, A. D., and Launder, B. E.: "The Near-Field Character of a Jet Discharged Through a Wall at 90 Degrees to a Main Stream", ASME paper 75-WA/HT-108, 1975.

[7] Pratap, V. S., and Spalding, D. B.: "Fluid Flow and Heat Transfer in Three-Dimensional Duct Flows", Int. J. Heat and Mass Transfer, Vol. 19, 1976, p. 1183.

[8] Briley, W. R., and McDonald, H.: "On the Structure and Use of Linearized Block ADI and Related Schemes"; SRA Report R78-3, 1978.

The Present Approach

The present approach is based on solution of the time-dependent three-dimensional compressible Navier-Stokes equations using an efficient implicit finite-difference scheme. Of the various implicit schemes one might consider, the most efficient appear to be the noniterative split linearized block implicit (LBI) schemes recently reviewed by Briley and McDonald (Ref. 8). Consistently split noniterative LBI schemes were first introduced in 1973 by Lindemuth and Killeen (Ref. 9) and Briley and McDonald (Ref. 10). Both of these methods employ consistent splittings (Peaceman-Rachford and Douglas-Gunn, respectively), but differ slightly in implementation of the linearization. The Douglas-Gunn procedure is a generalized method for deriving split forms of implicit schemes in multidimensions, and includes as a special case the two-dimensional Peaceman-Rachford splitting. The present application requires a scheme applicable in three dimensions, and thus the method of Briley and McDonald (Ref. 10) is being used. This method has been used previously to predict flow in rectangular ducts (Ref. 11), in three-dimensional combustors (Refs. 12 and 13) and about airfoils (Ref. 14).

Although the procedure is time consistent, the time step can be regarded as merely a computational artifice to obtain rapid convergence to

---

[9] Lindemuth, I., and Killeen, J.: "Alternating Direction Implicit Techniques for Two-Dimensional Magnetohydrodynamics Calculations", J. Computational Physics, Vol. 13, October, 1973, p. 181.

[10] Briley, W. R., and McDonald, H.: "An Implicit Numerical Method for the Multidimensional Compressible Navier-Stokes Equations", United Aircraft Research Laboratories Report, M911363-6, November , 1973.

[11] Briley, W. R., and McDonald, H.: "Solution of the Multidimensional Compressible Navier-Stokes Equations by a Generalized Implicit Method", J. Computational Physics, Vol. 24, August, 1977, p. 372.

[12] Gibeling, H. J., McDonald, H. and Briley, W. R.: "Development of a Three-Dimensional Combustor Flow Analysis, Volume I, AFAPL-TR-75-59, 1975.

[13] Gibeling, H. J., McDonald, H., and Briley, W. R.: "Development of a Three-Dimensional Combustor Flow Analysis, Volume II, AFAPL-TR-75-59, 1976.

[14] Shamroth, S. J., and Gibeling, H. J.: "The Prediction of the Turbulent Flow Field About An Isolated Airfoil", AIAA Paper 79-1543, 1979.

the steady state. Should the physical transients be of interest, time steps suitable for accurate determination of the temporal flow changes can be specified, but almost certainly these time steps would be smaller and more numerous (hence more costly) than those required to simply obtain rapid convergence to steady state with minimum computational expense. Further economy in the solution procedure is obtained through the use of zone embedding concepts, as discussed by McDonald and Briley (Ref. 15). Zone embedding limits the computational domain to the region of particular interest, and treats the remainder of the flow field either by means of interactive boundary conditions or by using a less complicated and more economical analysis, or both. Overall efficiency is improved since grid points are used only where they are absolutely necessary. In the present study, the computational domain is limited to the region near the coolant injection hole, and interactive boundary conditions consistent with the flow field upstream and downstream of injection are applied. For internal subsonic flows, this treatment permits the overall mass flow to adjust to the losses present in the coolant injection region, and permits disturbances and flow variations of a predetermined type (arising from the coolant flow pattern) to leave the computational region. Although the details of the interactive boundary conditions vary with each application, the zone embedding approach has been used successfully in several problems including combustor flows, flow past cascades of airfoils, and airfoil leading edge horseshoe vortex formation.

Results for a Model Problem

Since a determination of suitable interactive boundary conditions is a key element of the present investigation of the coolant injection problem, the first phase of the investigation consisted of performing test calculations using different interactive boundary conditions. For this purpose, a two-dimensional slot injection configuration was used. This test problem has

---

[15] McDonald, H., and Briley, W. R.: "Computational Fluid Dynamic Aspects of Internal Flows", AIAA Paper 79-1445, 1979.

most of the relevant features required for study of the interactive boundary conditions, since the only additional boundary conditions required for the three-dimensional problem are straightforward symmetry conditions. The two-dimensional model problem thus provides an understanding of the injectant-freestream interaction process without the added complexity or cost of full three-dimensional computations. The specific problem considered was flow between parallel plates with normal injection and a fully-developed inlet flow with Reynolds number of 100. The computational mesh consisted of eleven points in the normal direction and 19 points in the streamwise direction. A hyperbolic tangent transformation was used to concentrate grid points near the wall. In the streamwise direction, the high degree of resolution found necessary behind the region of injection was obtained using a hyperbolic sine transformation. The computational domain was five jet diameters in height and extended four diameters upstream of the injection and five diameters downstream. At the upstream boundary the boundary conditions were specified as follows: the normal velocity component was set to zero, the density was determined from the state equation for a perfect gas, and a total pressure profile was determined from the assumed fully-developed streamwise velocity distribution. With the total pressure specified, the average inlet velocity could vary, thus adjusting the inlet mass flux to account for flow losses caused by the injection. At the downstream boundary the static pressure was specified and the velocity components were extrapolated from the interior flow field. Along the outer boundary, symmetry conditions were specified and no-slip conditions were specified along the wall, together with extrapolation of density. Two computations were then carried out with different boundary conditions at the injection point.

The first calculation was performed with a prescribed uniform injection velocity equal to 0.1 of the freestream. This type of boundary condition does not allow the freestream flow to influence the injection rate or velocity profile. Streamwise velocity profiles for this calculation are shown in Fig. 1 at a location 1.5 D upstream of the injection and 2 D downstream. The interactive nature of the upstream boundary can be observed in this figure as the freestream velocity has been reduced to 0.94 from an estimated 1.0 due to losses present in the flow region. Also evident is a

tendency for the flow to separate, as can be observed by examining the shape of the downstream velocity profile near the wall, although separation did not occur in this case.

For the second calculation, the specified injection velocity was replaced by a specification of constant total pressure at the injection port. This boundary condition permits the jet mass flux and velocity distribution to interact freely with the freestream flow. Results of this calculation are shown in Fig. 2. The total pressure at the injection port was chosen to yield an estimated average injection velocity of about 0.1 times that of the freestream. It is again observed that the presence of the jet restricts the oncoming flow and that the flow downstream of the jet exhibits a tendency towards separation. Also present in this calculation is a nonconstant jet injection velocity profile, which results from use of the interactive boundary condition at the port. This type of nonuniform jet velocity is typical of that observed experimentally in three-dimensional configurations. The degree of nonuniformity for the two-dimensional configuration in Fig. 2 is less than that observed in discrete hole cooling experiments, but this is likely to be a consequence of the three-dimensional flow geometry, which permits the freestream to be deflected laterally around the coolant jet. Use of the interactive coolant inflow boundary condition has permitted the inflow velocity distribution to adjust to the behavior of the freestream surrounding the coolant jet. In addition, the specification of total pressure at the freestream inlet and static pressure at the exit plane has permitted the average mass flux within the flow passage to adjust to the viscous losses present, including those associated with the coolant jet. These results for the two-dimensional model problem are very encouraging, and these boundary conditions will be further developed and applied to the three-dimensional discrete hole coolant injection problem during the next phase of the investigation.

SUMMARY AND CONCLUDING REMARKS

An analysis and computational procedure for the discrete hole film cooling problem, based on numerical solution of the three-dimensional Navier-Stokes equations, has been initiated. The approach is based on numerical solution of the three-dimensional compressible Navier-Stokes equations, and utilizes a zone embedding procedure to limit the computational domain to the immediate vicinity of the coolant injection. The zone embedding is accomplished through the use of interactive boundary conditions at inflow and outflow boundaries, which are compatible with flow behavior outside the computational region. In this initial phase of the investigation, techniques for applying interactive boundary conditions were examined for a simplified model problem consisting of normal injection through a two-dimensional slot. The zone embedding approach was found to be successful in limiting the computational region, and the use of specified total pressure as an injection boundary condition was found to produce a nonuniform inlet velocity distribution consistent with interaction between the inlet jet and the freestream. During the next phase of the investigation, these zone embedding techniques will be refined and applied to the three-dimensional discrete hole film cooling configuration.

REFERENCES

1. Colladay, R. S. and Russell, L. M.: "Streakline Flow Visualization of Discrete-Hole Film Cooling With Normal, Slant, and Compound Angle Injection", NASA TN D-8248, September, 1976.

2. Bergeles, G., Gosman, A. D. and Launder, B. E.: "The Prediction of Three-Dimensional Discrete-Hole Cooling Processes: I-Laminar Flow", ASME Paper 75-WA/HT-109, 1975.

3. Crawford, M. E., Kays, W. M., and Moffat, R. J.: "Heat Transfer to a Full-Coverage Film-Cooled Surface with 30 Degree Slant-Hole Injection", Thermoscience Division, Dept. Mech. Engr. Stamford University, Report HMT-25, May, 1976.

4. Ericksen, V. L., Eckert, E. R. G., and Goldstein, R. J.: "A Model for Analysis of the Temperature Field Downstream of a Heated Jet Injected into an Isothermal Crossflow at an Angle of 90° ", NASA CR-72990, 1971.

5. Shamroth, S. J. and McDonald, H.: "Calculations of Film Cooling, Wall Transpiration, and Heat Transfer with a Finite-Difference Boundary Layer Procedure", United Aircraft Research Laboratories Report N212634-1, January, 1973.

6. Bergeles, G., Gosman, A. D., and Launder, B. E.: "The Near-Field Character of a Jet Discharged Through a Wall at 90 Degrees to a Main Stream", ASME paper 75-WA/HT-108, 1975.

7. Pratap, V. S., and Spalding, D. B.: "Fluid Flow and Heat Transfer in Three-Dimensional Duct Flows", Int. J. Heat and Mass Transfer, Vol. 19, 1976, p. 1183.

8. Briley, W. R., and McDonald, H.: "On the Structure and Use of Linearized Block ADI and Related Schemes"; SRA Report R78-3, 1978.

9. Lindemuth, I., and Killeen, J.: "Alternating Direction Implicit Techniques for Two-Dimensional Magnetohydrodynamics Calculations", J. Computational Physics, Vol. 13, October, 1973, p. 181.

10. Briley, W. R., and McDonald, H.: "An Implicit Numerical Method for the Multidimensional Compressible Navier-Stokes Equations", United Aircraft Research Laboratories Report, M911363-6, November, 1973.

11. Briley, W. R., and McDonald, H.: "Solution of the Multidimensional Compressible Navier-Stokes Equations by a Generalized Implicit Method", J. Computational Physics, Vol. 24, August, 1977, p. 372.

12. Gibeling, H. J., McDonald, H. and Briley, W. R.: "Development of a Three-Dimensional Combustor Flow Analysis", Volume I, AFAPL-TR-75-59, 1975.

REFERENCES (Continued)

13. Gibeling, H. J., McDonald, H., and Briley, W. R.: "Development of a Three-Dimensional Combustor Flow Analysis, Volume II, AFAPL-TR-75-59, 1976.

14. Shamroth, S. J., and Gibeling, H. J.: "The Prediction of the Turbulent Flow Field About An Isolated Airfoil", AIAA Paper 79-1543, 1979.

15. McDonald, H., and Briley, W. R.: "Computational Fluid Dynamic Aspects of Internal Flows", AIAA Paper 79-1445, 1979.

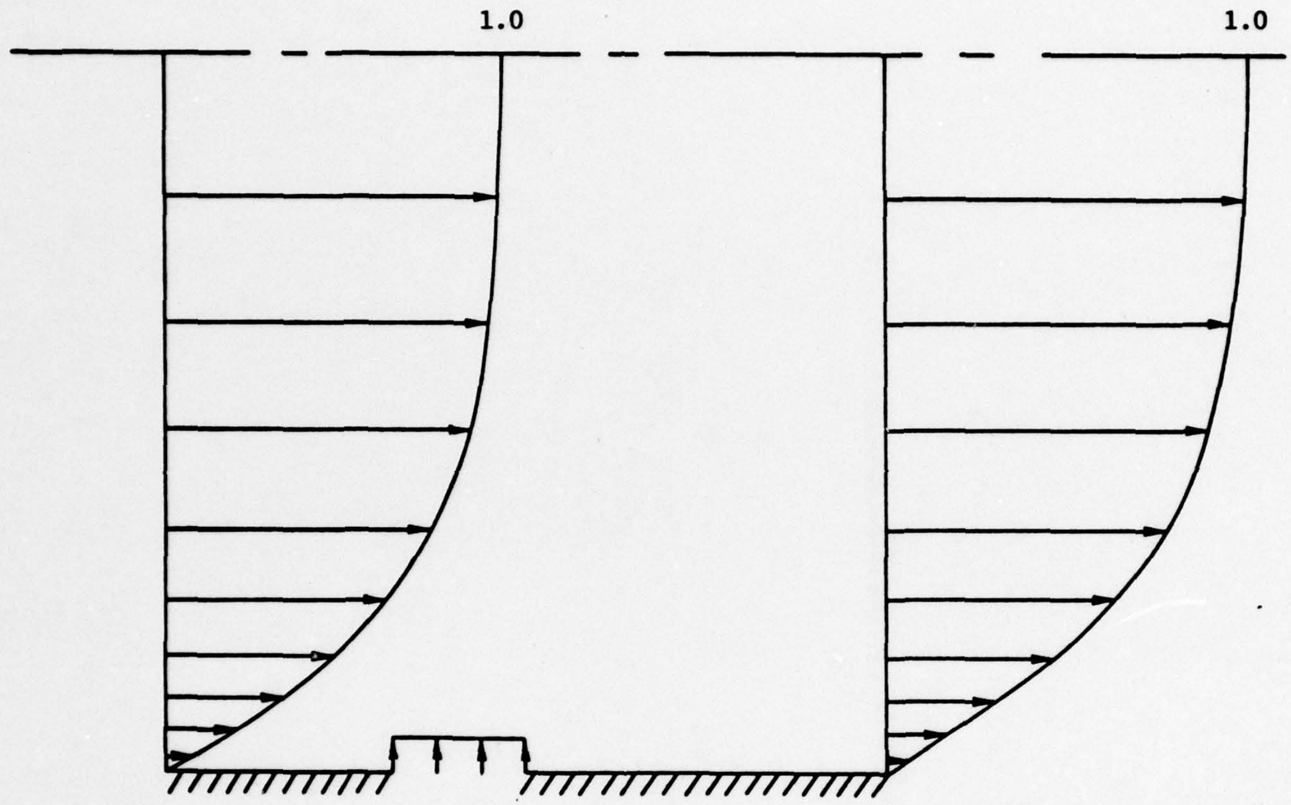


Figure 1. Computed velocity profiles for coolant injection model problem. Re = 100, M = 0.1, U_J/U_m = 0.1, constant velocity normal injection is specified.

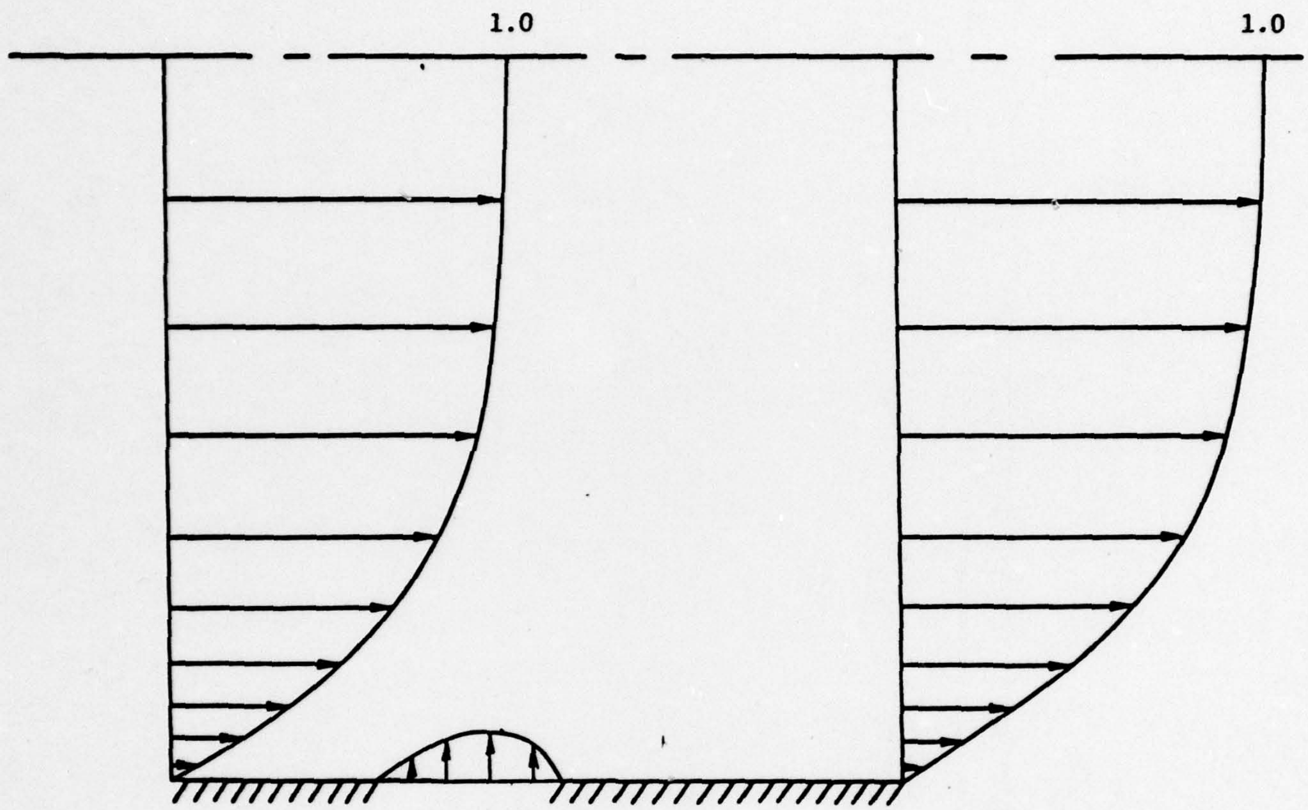


Figure 2. Computed velocity profiles for coolant injection model problem. Re = 100, M = 0.1, $\int_a^b u_J dx/U_m = 0.1$, constant total pressure normal injection is specified.